NASA/TM—2001-211132 IECEC2001–CT–38

CFD Modeling of Free-Piston Stirling Engines

Mounir B. Ibrahim
Cleveland State University, Cleveland, Ohio

Roy C. Tew, Jr.
Glenn Research Center, Cleveland, Ohio

Zhiguo Zhang
Cleveland State University, Cleveland, Ohio

David Gedeon
Gedeon Associates, Athens, Ohio

Terrence W. Simon
University of Minnesota, Minneapolis, Minnesota

September 2001

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA Access Help Desk at 301–621–0134

- Telephone the NASA Access Help Desk at 301–621–0390

- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

NASA/TM—2001-211132 IECEC2001–CT–38

CFD Modeling of Free-Piston Stirling Engines

Mounir B. Ibrahim
Cleveland State University, Cleveland, Ohio

Roy C. Tew, Jr.
Glenn Research Center, Cleveland, Ohio

Zhiguo Zhang
Cleveland State University, Cleveland, Ohio

David Gedeon
Gedeon Associates, Athens, Ohio

Terrence W. Simon
University of Minnesota, Minneapolis, Minnesota

Prepared for the
36th Intersociety Energy Conversion Engineering Conference
cosponsored by the ASME, IEEE, AIChE, ANS, SAE, and AIAA
Savannah, Georgia, July 29–August 2, 2001

National Aeronautics and
Space Administration

Glenn Research Center

September 2001

Acknowledgments

This work was conducted under a NASA Grant to Cleveland State University, NAG3–2482. Professor Ibrahim, Cleveland State University, Professor Simon, University of Minnesota, and David Gedeon, Gedeon Associates, are Co-Principle Investigators. The Thermo-Mechanical Branch, Power and On-Board Propulsion Division, NASA Glenn Research Center, under the leadership of Mr. Richard Shaltens, Branch Chief, is greatly acknowledged.

This report is a formal draft or working paper, intended to solicit comments and ideas from a technical peer group.

This report contains preliminary findings, subject to revision as analysis proceeds.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at http://gltrs.grc.nasa.gov/GLTRS

CFD MODELING OF FREE-PISTON STIRLING ENGINES

Mounir B. Ibrahim
Cleveland State University
Cleveland, Ohio 44115

Roy C. Tew, Jr.
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Zhiguo Zhang
Cleveland State University
Cleveland, Ohio 44115

David Gedeon
Gedeon Associates
Athens, Ohio 45701

Terrence W. Simon
University of Minnesota
Minneapolis, Minnesota 55455

ABSTRACT

NASA Glenn Research Center (GRC) is funding Cleveland State University (CSU) to develop a reliable Computational Fluid Dynamics (CFD) code that can predict engine performance with the goal of significant improvements in accuracy when compared to one-dimensional (1-D) design code predictions. The funding also includes conducting code validation experiments at both the University of Minnesota (UMN) and CSU. In this paper a brief description of the work-in-progress is provided in the two areas (CFD & Experiments). Also, previous test results are compared with computational data obtained using: 1) a 2-D CFD code obtained from Dr. Georg Scheuerer and further developed at CSU and 2) a multi-dimensional commercial code CFD-ACE+. The test data and computational results are for: 1) a gas spring and 2) a single piston/cylinder with attached annular heat exchanger. The comparisons among the codes are discussed. The paper also discusses plans for conducting code validation experiments at CSU & UMN.

INTRODUCTION

The type of flows that are normally encountered in Stirling Engines are: 1) Unsteady (Oscillatory Flow/ Oscillatory Pressure), 2) 2-D/3-D Flows (Complex Geometry), 3) Compressible (low Mach Number), Laminar/Transition/ Turbulent, 4) Unsteady Conduction/Convection Heat Transfer, 5) Sudden changes in cross-section, 6) Isothermal/adiabatic boundary conditions, and 7) Single phase with no chemical reaction. On the other hand the losses encountered in Stirling engines (Tew 1988 & Tew and Geng 1992) are: 1) Gas spring and working space hysteresis losses, 2) Gas-to-heat exchanger heat transfer inefficiencies, 3) Viscous (pressure drop) losses, 4) Appendix gap losses (shuttle and pumping), 5) Mixing losses due to mixing of gases at different temperatures, 6) Conduction losses from the hot- to the cold-end of the engine. In addition, there are losses due to differences in flow distribution: a) from one heat exchanger flow passage to the other and/or b) across the regenerator (1-D codes assume uniform flow).

A successful CFD code that simulates the engine adequately should capture the characteristics of the flow and the type of losses described above.

Stirling One-Dimensional Codes

During the late 70's and early 80's, NASA developed it's own finite-difference Stirling engine performance code for use in monitoring the work of its Stirling engine contractors (Tew, et. al.1978 & Tew 1983). Later NASA gained unlimited rights to the MTI developed HFAST (Huang 1993) harmonic code, via a contract with MTI. NASA also purchased the GLIMPS Stirling engine code from David Gedeon. GLIMPS (now developed into Sage, Gedeon 1995) has been the primary design tool used by the Stirling Technology Co. (STC) in recent years. HFAST and GLIMPS (now Sage) were both more time-efficient and user-friendly than the NASA's finite-difference code; and they were both being used to design real hardware.

Tew and Geng,1992 showed comparisons of the losses calculated by GLIMPS and HFAST for the 12.5 kWe Component Test Power Converter (CTPC). Although the overall engine power and efficiency predictions were quite close, there were substantial differences in some of the loss calculations. For example GLIMPS calculations suggested that cylinder hysteresis power losses were about 10% of the indicated power while HFAST calculated a much smaller value (~3%) for this loss. HFAST predicted larger viscous and mixing losses than GLIMPS, and so the overall engine performance predictions were similar. Geng and Tew, 1992 showed additional HFAST/GLIMPS comparisons, both "calibrated" and "uncalibrated", for the 1.2 kW indicated power RE-1000 engine and the 14 kW indicated power Space Power Demonstrator Engine (SPDE). GLIMPS also calculated that cylinder hysteresis losses were about 10% of the indicated power for these engines.

The Sage commercial code (Gedeon 1995) was introduced about six years ago. The Sage Stirling-cycle modeling software is the latest in a line of commercial software developed by Gedeon Associates. It is a direct descendant of the GLIMPS software, which was used widely within the Stirling industry for nearly ten years. Sage introduced a drag-and-drop visual interface where a user could assemble complete machines from standard components, such as pistons, cylinders, heat-exchangers, etc. Sage also introduced an interactive optimization capability built into the visual interface.

CAST and Modified CAST Code

A CFD code, CAST, received by Dr. Mounir Ibrahim from Dr. Georg Scheuerer, was developed by Peric and Scheuerer 1989. CAST, an acronym for Computer Aided Simulation of Turbulent Flows, is a computer program which uses the finite volume method for predicting two-dimensional flow and heat transfer phenomena. The CAST code is written in FORTRAN IV. CAST is similar in structure to existing fluid flow prediction procedures like TEACH (Gosman and Ideriah, 1976) and TEAM (Huang and Lewchziner, 1983); it differs from those codes in the following ways (1) it has a co-located variable arrangement, (2) it has a different discretization scheme, (3) the solution algorithms are different for the linear equation systems arising from the discretization, and (4) the pressure-velocity coupling is adapted to the co-located variable storage. CAST solves the Navier-Stokes equations. For turbulent flows, the Reynolds-averaged Navier-Stokes equations are solved in connection with the k-ε, two-equation turbulence model of Launder and Spalding.

The CAST code was used by several Cleveland State University (CSU) graduate students working under the direction of Dr. Mounir Ibrahim for two-dimensional modeling of “Stirling machine like” components. This work was conducted under grants from NASA Glenn Research Center (which was NASA Lewis at the time of the work) in the period from 1989 to1994.

Below is a summary of the code development topics:
1) Oscillating inlet & outlet velocity conditions, 2) Sudden change in channel cross section, i.e. sudden expansion in one half of the cycle and sudden contraction in the other half, 3) Different Low-Reynolds Number, k-ε, turbulent models, 4) A model for laminar-transition-turbulent in a pipe flow, 5) Incompressible-flow with moving boundaries and, finally, 6) Compressible-flow with moving boundaries.

This work is summarized below:
Ahn 1992, added a low-Reynolds-Number k-ε turbulence model to CAST for purposes of conducting studies of laminar to turbulent transition in pipe flows. This modified CAST code was used in cooperation with experimental studies at the University of Minnesota to help determine that upon relaminarization of turbulent pipe flow over a periodic cycle, the laminar flow at the beginning of the new cycle had attained the characteristics of a steady, uniform velocity distribution (in the radial direction).

Hashim, 1992, configured the CAST code to simulate oscillating flow and heat transfer in parallel-plate channels with a sudden change in cross-section. The flow was assumed to be laminar and incompressible with the inflow velocity uniform over the channel cross-section but varying sinusoidally with time. Ibrahim et. al., 1992, using the computational results, were able to determine instantaneous friction factors and heat transfer coefficients for laminar oscillating flow between parallel plates with a sudden change in cross-section. It was found that instantaneous friction factors and heat transfer coefficients deviated substantially from the steady-state values for the same dimensionless flow parameters.

Kannapareddy, 1993, used CAST to model laminar, incompressible, oscillatory flow in the heater, regenerator and cooler of the NASA Stirling Space Power Research Engine (SPRE). The heater and cooler tubes were modeled as circular pipes with isothermal walls; the regenerator was modeled as two-parallel plates. Although the flow was oscillatory, when the flow was into the heater or cooler, the input temperatures were assumed to be constant. The study was carried out for a wide range of Maximum Reynolds numbers (based on maximum flow velocity), Valensi numbers (non-dimensional frequencies), and relative amplitudes of fluid displacement in the component of interest; the ranges chosen were based on the operating ranges of the NASA SPRE. The instantaneous friction factor, wall heat flux and heat transfer coefficient were examined. It was concluded that the friction factor and heat transfer coefficients are larger under oscillatory flow conditions for larger Valensi numbers (i.e., larger non-dimensional frequencies). Also the thermal efficiency of the heat exchangers decreased for the lower fluid displacement values and the steady-state definition for the heat transfer coefficient did not appear valid for use with oscillatory-flow ranges studied. A concise presentation of this work was reported by Ibrahim and Kannapareddy, 1992. In work that was not documented, Kannapareddy modified CAST to incorporate a moving boundary model to allow simulation of piston motion. These modifications were based upon information in Ferziger and Peric, 1997.

Bauer, 1993, used CAST with the low-Reynolds number k-ε, turbulence model option (Lam-Bremhorst form), to assist Ibrahim et al., 1994, in developing an empirical transition model that could be used to predict when in the University of Minnesota (UMN) periodic-pipe-flow experiments (Qiu and Simon, 1994) transition laminar to turbulent flow would occur. The empirical transition model was used to activate the turbulence model at the appropriate time within the cycle for a given axial location in the tube. This analytical technique for modeling unsteady flow and heat transfer in Stirling engine heater and cooler tubes (Simon, et al. 1992, and Ibrahim, et al., 1994) is now an essential component of the 1-D system simulation code Sage (at that time called GLIMPS). The tests used for this development showed important features of oscillatory flow, which led to enhanced understanding of the mechanisms for losses in Stirling systems. Such features include transition from the laminar flow to turbulence in each cycle of oscillation and the rapid dissipation of the turbulence, which remains from one cycle as a result of the extremely strong axial straining of the fluid upon temporal acceleration at the beginning of the following cycle. This collaborative effort showed the value of combining computational results with experimental finding for better understanding of Stirling engine flow and heat transfer processes.

In searching for a compressible flow simulation techniques suitable for simulation of Stirling machines, the literature was first searched for papers relating to “low Mach number compressible flow.” Many such papers were obtained. For example, papers by Pletcher and Chen, 1993, Chen and Pletcher, 1991, Sesterhenn, Muller et al.1993, Turkel, 1987, Weiss and Smith, 1993, and Horibata, 1992 were reviewed. These papers appeared to take the approach of starting with a fully compressible model (including acoustics) and making suitable modifications to the solution technique so that the calculations could be extended to low Mach number flow. If any of these approaches had been taken to modify the CAST code, it appeared that it would be almost equivalent to starting over and developing a new code. And, these approaches did not appear to offer the time benefits that could be achieved by complete elimination of acoustic phenomena.

For all practical Stirling engines with which these authors have familiarity, flow occurs at very low Mach numbers (usually much less than 0.1). However, compressibility is very important because volume changes due to piston motion in an enclosed volume produce large changes in the engine pressure level and density over the cycle. Due to the compact nature of Stirling machines and the moderate frequencies of operation (< 125 Hz), spatial pressure variations at any given time are relatively small so that spatial density variations are due almost entirely to spatial temperature variations. Calculations have shown that it appears to be a good approximation to assume that the speed of sound is infinitely fast inside most Stirling machines. Some authors, such as Organ, 1992, appear to contest this assumption

The literature search continued for a compressible flow approach based on the assumption that acoustics are negligible. An excellent paper explaining such an approach was found in the 1997 Journal of Fluid Mechanics entitled “A model of unsteady subsonic flow with acoustics excluded” by A.T. Fedorchenko, 1997. The derivations include a set of equations for simulating subsonic flow of a heat-conducting, viscous gas. The simplifications in the Navier-Stokes equations used to eliminate acoustics include: (1) Pressure at any time and spatial location is split into a mean pressure level that varies only with time and a delta-pressure relative to a reference location that varies with spatial position and time; (2) The pressure appearing in the equation of state (for example, the ideal gas equation of state) is the mean pressure that varies only with time. Therefore, from the equation of state, density is a function only of the mean pressure level and the temperature (which varies with spatial position and time); and (3) The pressure appearing in the momentum equations is just the delta-pressure relative to the reference location.

Thus, Fedorchenko’s paper provided the basis for modifying the CAST code. A technique from an early finite-difference one-dimensional flow non-acoustic code (Tew, Jefferies and Miao, 1978, Tew, 1983) was incorporated to ensure an accurate mass balance. It was not necessary to modify the incompressible form of the SIMPLE algorithm used in CAST since density does not vary with the delta-pressure from the reference location. More details are available in (Tew & Ibrahim, 2001)

The modified CAST can now be used to simulate: (1) fixed-boundary, incompressible flow with an inlet and an outlet boundary, (2) incompressible flow with a moving boundary (piston, for example) with one inlet/outlet, or (3) compressible, non-acoustic, flow in a completely enclosed volume with a moving boundary.

CFD-ACE+ Code

CFD-ACE+ is a commercial CFD code product of CFD Research Corporation. It is a set of computer programs for multi-physics computational analysis. The programs provide an integrated geometry and grid generation module, a graphical user interface for preparation of the model, a computational solver for performing the simulation, and an interactive visualization program for examination and analysis of the simulation results. The standard CFD-ACE+ package includes the following applications: geometry and grid generation, advanced polyhedral solver and a post processor.

CFD-ACE+ has been used for design and analysis of a wide variety of industrial applications. It can model 2-D or 3-D geometries and includes acoustics.

ANALYSIS

For this paper analyses were performed using CFD-ACE+ and the Modified CAST code for gas spring and piston/cylinder/heat exchanger (or two-space) cases (See Appendix A for test rig dimensions originating from Kornhauser, 1989):

(1) Gas Spring case: The computational domain is 2-D and axisymmetric, with minimum dimensions of 0.0762 (x-direction) *0.0254(m) Piston stroke is 0.0762 m. Also 86*40 grids and 200 time-steps/cycle were used.

(2) Two-space case: The computational domain is 2-D and axisymmetric, with dimensions of ~0.445 (x-direction) *0.0254(m). The piston stroke was also 0.0762 m for this case. 148*48 grids and 960 time-steps/cycle were used.

Helium was used as the working fluid, with the standard k-ε, turbulence model and upwind finite differencing scheme.

RESULTS

Gas Spring Hysteresis Losses

For a gas spring, the hysteresis loss is the work that is dissipated by the spring per cycle at steady operating conditions; it’s also equal to the heat generated in and transferred out of the spring. A good way to compare computational and measured hysteresis losses is via plots of dimensionless work as a function of oscillating flow Peclet number. Dimensionless work and oscillating flow Peclet numbers are defined, respectively, as follows:

$$\hat{W}_{loss} = \frac{\oint P\,dV}{P_o V_o \left(\frac{P_a}{P_o}\right)^2 \left(\frac{\gamma - 1}{\gamma}\right)} \qquad (1)$$

$$Pe_\omega = \frac{\rho_o c_p \omega D_h^2}{4\lambda} \qquad (2)$$

Where:

c_p specific heat at constant pressure,

D_h hydraulic diameter (=4 x wetted area/wetted perimeter),

P mean spatial pressure,

P_a amplitude of mean spatial pressure

P_o arithmetic mean of max. and min. mean spatial pressures

V volume

V_o arithmetic mean of maximum and minimum volumes

$\hat{W}_{loss}$ non-dimensional work or hysteresis loss

Greek letters

γ ratio of specific heats of fluid

λ molecular thermal conductivity

ρ density, fluid mass per unit volume

ω angular velocity (rad/sec)

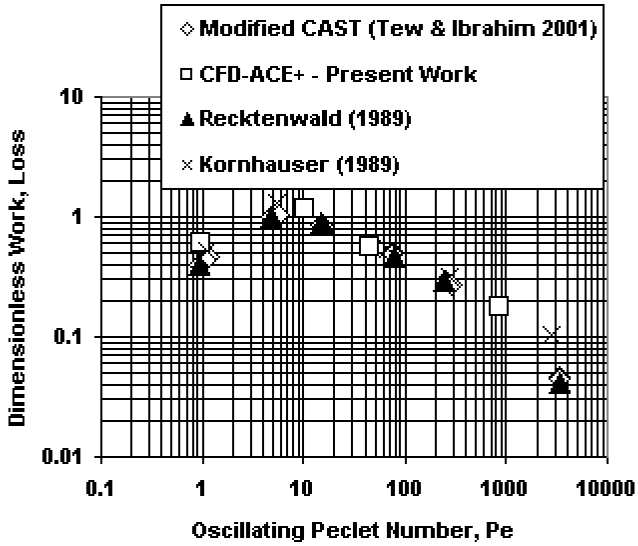

Figure 1: Comparison Among CFD Codes for Dimensionless Losses & Kornhauser's (1989) Experimentally Derived Losses.

Figure 1 shows the dimensionless work plotted versus the Peclet number for the computations of CFD-ACE+, Modified CAST and Recktenwald (1989) together with Kornhauser's (1989) data. Only four data points were chosen for CFD-ACE+ over a range of Pe from 0.97 to 843. The CFD-ACE+ predictions fit very well with all the other results presented in Figure 1. More detailed comparisons between the computational results and experimental data are given in a companion paper (Tew and Ibrahim 2001).

Two-Space Test Rig Data and Calculation Comparisons

Figures 2a and 2b show CFD results obtained from CFD-ACE+ and Modified CAST code respectively. The data are for the temperature difference between the wall and the center of the heat exchanger at the following different axial locations: entrance of the heat exchanger and 1/16, 1/8, 1/4 & 1/2 of the annulus total length, measured from the entrance.

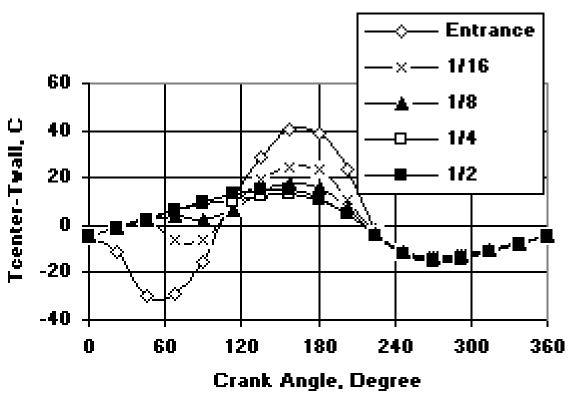

Figure2a: CFD-ACE+ Calculations of Temperature Difference from Heat Exchanger Center to Wall (148x48 grids, 960 time steps/cycle). For Comparison with Kornhauser (1989) Exp. Data: Run #12071539, 201.7 RPM, 1.008 MPa Mean Pressure.

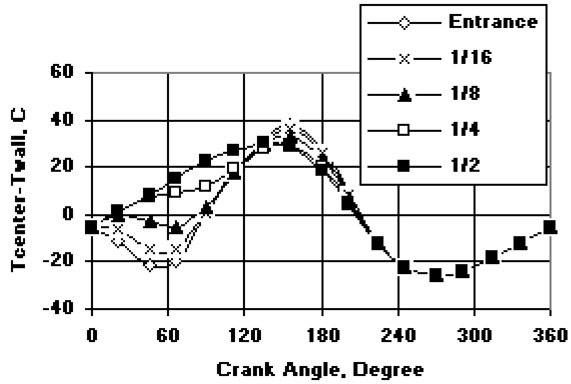

Figure 2b: Modified CAST Calculations of Temperature Difference from Heat Exchanger Center to Wall (34x20 grids, 120 time steps/cycle). For Comparison with Kornhauser (1989) Exp. Data: Run #12071539, 201.7 RPM, 1.008 MPa Mean Pressure.

Upon comparing Figures 2a & 2b, the two codes CFD-ACE+ & Modified CAST behave in a similar way to the experimental data (see Tew & Ibrahim 2001) with some differences in the following: 1) Modified CAST predicts maximum temperature differences (center/wall), except at the entrance, larger than CFD-ACE+ and closer to the experimental data, 2) Also Modified CAST shows effects of the axial location (up to 180 Degrees-Crank-Angle) more like the experimental data trend than CFD-ACE+, 3) Both codes show results independent of the axial location beyond 230 degrees-crank-angle, unlike the experimental data.

Figures 3a and 3b show CFD results obtained from CFD-ACE+ and the Modified CAST code, respectively. The results are for heat transfer per unit area at the following different locations in the heat exchanger: entrance of the heat exchanger and 1/16, 1/8, 1/4 & 1/2 of the annulus total length measured from the entrance.

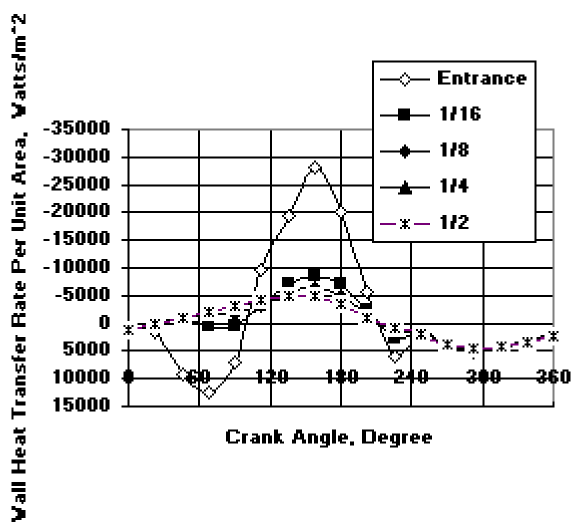

Figure 3a: CFD-ACE+ Calculations of Heat Transfer per Unit Area at Different Locations in the Heat Exchanger Relative to Entrance to Cylinder (148x48 grids, 960 time steps/cycle). For Comparison with Kornhauser (1989) Exp. Data: Run #12071539, 201.7 RPM, 1.008 MPa Mean Pressure.

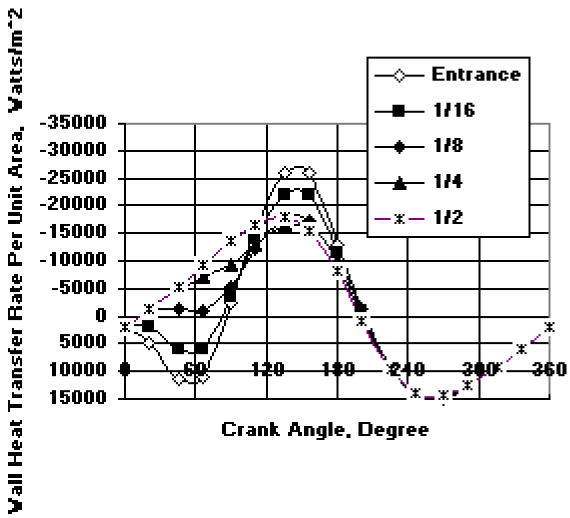

Figure 3b: Modified CAST Calculations of Heat Transfer per Unit Area at Different Locations in the Heat Exchanger Relative to Entrance to Cylinder (34x20 grids, 120-time steps/cycle). For Comparison with Kornhauser (1989) Exp. Data: Run #12071539, 201.7 RPM, 1.008 MPa Mean Pressure.

Again, upon comparing Figures 3a & 3b, the two codes CFD-ACE+ & Modified CAST behave in a similar way to the experimental data (see Tew & Ibrahim 2001) with some differences in the following: 1) Modified CAST shows the effect of the axial location more like the experimental data trend, however, 2) except at the entrance, the CFD-ACE+ heat transfer/unit area is closer to the experimental data than Modified CAST.

The results obtained at this point emphasize the fact that in order to obtain a reliable CFD simulation for the Stirling engine, there is a need for accurate experimental data. The section below describes the test facilities at CSU & UMN and the plan for running code validation experiments.

CODE VALIDATION EXPERIMENTS

CSU-SLRE Test Rig The CSU Stirling-Laboratory-Research-Engine (SLRE) derives from a two-piston Stirling-engine test rig developed by NASA Jet Propulsion Laboratory in 1982 (Hoehn, 1982). A special test section is being designed to be bolted between the two existing piston-cylinder flanges on the rig. The pistons will drive atmospheric-pressure air or helium through the test section at Reynolds and Valensi numbers representative of typical Stirling engine practice. Within the test section will be various passages representative of regions of the Stirling engine where flow area changes abruptly or flow velocity changes direction. This may include the ports leading to the compression space or the transition between tubes or rectangular passages leading to the expansion space. In the test rig, the passages will be two dimensional, either axial (x, r) or rectangular (x, y), so that the flows can be compared directly with 2-D codes.

Flow measurement will be either by LDV (Laser-Doppler Velocimetry) or PIV (Particle Imaging Velocimetry), through transparent walls in the test section. Both methods require tracer particles within the gas flow. LDV measures individual tracer particle velocities one after another as it encounters them within its measurement volume, whereas PIV measures trajectories of many particles simultaneously within a measurement plane. PIV is the preferred method because it provides flow field rather than single point information. For best compatibility with laser measurements, test sections with flat walls are best. A cylindrical test section requires optical correction for measurements of other than the axial component of velocity.

The following illustration shows a cylindrical test section comprising a transparent acrylic tube with an insert that produces an annular flow passage leading to an abrupt expansion at the center of the section. It will probably not be built, due to concerns about laser measurements through the circular wall. A rectangular test section has yet to be designed.

UMN Test Rig

The University of Minnesota test rig is an oscillatory flow facility, which is scaled up in size and down in frequency so the actual system is dynamically simulated. The drive section of the facility is a simple piston cylinder arrangement. The piston is driven by a scotch yoke arrangement so that its motion is a sinusoidally oscillating flow with a time mean bulk velocity of zero. The dimensionless amplitude and frequency match those of the engine through the Reynolds number based on peak velocity of the cycle and the Valensi number. This facility has been used for Stirling engine development under the SPRE Program (Qiu and Simon 1994). The facility is larger than the CSU-SLRE engine, which allows detailed measurements and clear visualization. It cannot match the engine frequency and must rely on dynamic similarity. The test section is a representation of a section of an engine. It consists of a cylindrical extension of the piston-cylinder region of the drive section and a radial channel between two disks, one of which is attached to that cylinder. It represents flow passages that could be found in either of the two ends of the engine, cold or hot section, in which the flow is driven

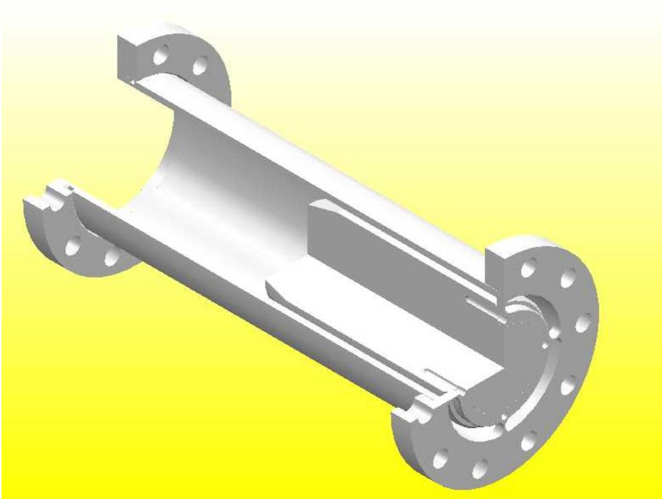

Figure 4 A Cylindrical Test Section Comprising a Transparent Acrylic Tube with an Insert, designed for the CSU-SLRE Test Rig.

axially, then radially, as it is pumped from the compression or expansion space radially outward toward the regenerator, then reverses. Flow through geometries of this type seems to not be computed well and there seems to be a need to improve turbulence closure and flow transition modeling. Measurements will include unsteady velocities, resolved in space and time. Further documentation includes visualization and measurement of unsteady flow separation and reattachment through the abrupt flow geometry change regions. Instruments which may be brought to bear on the problem are hot-wire anemometry, laser-Doppler anemometry and flow visualization with smoke, tufts and neutrally buoyant helium bubbles. Time records of the measurements will be made then decomposed according to time within the cycle and ensemble-averaged.

CONCLUDING REMARKS

Comparisons were made between the commercial CFD-ACE+ code and compressible non-acoustic calculations using Modified CAST, developed at CSU. The comparison was made for gas spring as well as two-space rig experimental data. The two codes predicted similar trends to the experimental data for, Work-Loss (gas spring), temperature differences (wall-center in the heat exchanger annulus) and wall heat flux per unit area. However, the two codes showed significant differences under certain conditions for the two-space test rig data. The reasons will be explored further. This work emphasizes the need for conducting accurate code validation experiments for oscillatory flow as it occurs in current Stirling engines.

REFERENCES

Ahn, K. and Ibrahim, M.B. (1992), "Laminar/Turbulent Oscillating Flow in Circular Pipes", Int. J. of Heat and Fluid Flow, Vol. 13, No. 4, pp 340-346.

Bauer, C. (1993), "Low-Reynolds Number k-epsilon and Empirical Transition Models for Oscillating Pipe Flow and Heat Transfer", M. S. Thesis, Cleveland State University.

Chen, K.-H. and R. H. Pletcher (1991), "Primitive Variable, Strongly Implicit Calculation Procedure for Viscous Flows at All Speeds", AIAA Journal, Vol. 29, No. 8.

Fedorchenko, A. T. (1997), "A Model of Unsteady Subsonic Flow with Acoustics Excluded", J. Fluid Mechanics, Vol, 334, pp. 135-155.

Ferziger, J. H. and Peric, M. (1997), "Computational Methods for Fluid Dynamics", Springer-Verlag.

Gedeon, D. (1995), *Sage: Object-Oriented Software for Cryocooler Design*, in*: Cryocoolers 8*, Edited by R.G. Ross, Jr., Plenum Press, pp. 281—292.

Geng, S. M. and Tew, R. C. (1992), "Comparison of GLIMPS and HFAST Stirling Engine Code Predictions with Experimental Data, NASA TM-105549.

Hashim, W. (1992), "Oscillating Flow and Heat Transfer in Channels with Sudden Change in Cross Section", Master's Thesis, Cleveland State University.

Hoehn, F.W. (1982)*, Stirling Laboratory Research Engine*, JPL Publication 82-13.

Huang, S.C. (1993), "HFAST Version 2.0 Analysis Manual", Prepared under NASA Contract Number NAS3-25330.

Ibrahim, M., Hashim, W., Tew, R. C. and Dudenhoefer, J. E. (1992), "Heat Transfer in Oscillating Flows with Sudden Change in Cross Section", NASA TM-105692.

Ibrahim, M.B. and Kannapareddy, M. (1994)"Computational Heat Transfer Analysis for Oscillatory Channel Flows" First ISHMT-ASME Heat and Mass Transfer Conference, Bombay, India.

Ibrahim, M., Bauer, C., Simon, T. and Qiu, S. (1994), "Modeling Oscillatory Laminar, Transitional, and Turbulent Channel Flows and Heat Transfer", International Heat Transfer Conference, London, England.

Kannapareddy, M. R. (1993), "Numerical Thermal Analysis of Heat Exchangers for the Stirling Engine Applications", M. S. Thesis, Cleveland State University.

Kornhauser, A. A. (1989), "Gas-Wall Heat Transfer During Compression and Expansion", Massachusetts Institute of Technology S. D. Thesis.

Peric, M. and Scheuerer, G.(1989), "CAST - A Finite Volume Method for Predicting Two-Dimensional Flow and Heat Transfer Phenomena", GRS - Technische Notiz SRR-89-01.

Pletcher, R.H. and Chen, K.-H. (1993), "On Solving the Compressible Navier-Stokes Equations for Unsteady Flows at Very Low Mach Numbers", AIAA 93-3368 and NASA TM-106380.

Qiu, S. and Simon, T. W. (1994), "Measurements of Heat Transfer and Fluid Mechanics Within an Oscillatory Flow in a Pipe", Winter ASME Annual Meeting.

Recktenwald, G. W. (1989), "A Study of Heat Transfer Between the Walls and Gas Inside the Cylinder of a Reciprocating Compressor", University of Minnesota Ph. D. Thesis.

Tew, R.C. Jr. and Ibrahim, M.B. (2001), "Study of Two-Dimensional Compressible Non-Acoustic Modeling of Stirling Machine Type Components," Paper IECEC 2001-CT-27, Proceedings of 36th Intersociety Energy Conversion Engineering Conference, July 29-August 2, 2001, Savannah, GA

Tew, Roy C. , Jr. (1988), "Overview of Heat Transfer and Fluid Flow Problem Areas Encountered in Stirling Engine Modeling", NASA TM-100131.

Tew, R. C., Jr. and Geng, S. M. (1992) , "Overview of NASA Supported Stirling Thermodynamic Loss Research", NASA TM-105690.

Tew, R. C., Jr., Jefferies, K. and Miao, D. (1978), "A Stirling Engine Computer Model for Performance Calculations", DOE/NASA/1011-78/24, NASA TM-78884.

Tew, Roy C. Jr. (1983), "Computer Program for Stirling Engine Performance Calculations", DOE/NASA/51040-42, NASA TM-82960.

Sesterhenn, Jorn, et. al. (1993), "Flux-Vector Splitting for Compressible Low Mach Number Flow", Computes Fluids, Vol. 22, No. 4/5, pp. 441-451.

APPENDIX

Gas Spring & Two-Space Test Rig Dimensions

Table A1: Gas Spring Dimensions

Physical Quantity	*Symbol*	*Value*
Cylinder Bore (Diam.)	D	50.80 mm (2 in.)
Piston Stroke	S	76.2 mm (3 in.)
Volume Ratio	r_V	2.0

Table A2: Two-Space Test Rig Dim. (Physical & Simulated)

Physical Quantity	*Physical Value*	*Simulation Value*
Cylinder Bore	50.80 mm (2 in)	50.80 mm (2 in)
Piston Stroke	76.20 mm (3 in)	76.20 mm (3 in)
Volume Ratio	2.0	2.0
Annulus O.D.	44.5 mm (1.75 in)	50.80 mm (2 in)
Annulus I.D.	39.4 mm (1.55 in)	46.4 mm (1.83 in)
Annulus Gap	2.5 mm (0.10 in)	2.2 mm (0.09 in)
Annulus Length	445 mm (17.5 in)	445 mm (17.5 in)
Min Pist/Head Clr.	2.9 mm (0.11 in)	2.9 mm (0.11 in)

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)

2. REPORT DATE
September 2001

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
CFD Modeling of Free-Piston Stirling Engines

5. FUNDING NUMBERS
WU–755–1A–14–00

6. AUTHOR(S)
Mounir B. Ibrahim, Roy C. Tew, Jr., Zhiguo Zhang,
David Gedeon, and Terrence W. Simon

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
John H. Glenn Research Center at Lewis Field
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E–12981

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM—2001-211132
IECEC2001–CT–38

11. SUPPLEMENTARY NOTES
Prepared for the 36th Intersociety Energy Conversion Engineering Conference cosponsored by the ASME, IEEE, AIChE, ANS, SAE, and AIAA, Savannah, Georgia, July 29–August 2, 2001. Mounir B. Ibrahim and Zhiguo Zhang, Mechanical Engineering Department, Fenn College of Engineering, Cleveland State University, 1960 East 24th Street, Cleveland, Ohio 44115–2425; Roy C. Tew, Jr., NASA Glenn Research Center; David Gedeon, Gedeon Associates: Sage Stirling, Pulse Tube, and Thermoacoustic Software, 16922 South Canaan Road, Athens, Ohio 45701; and Terrence W. Simon, Thermodynamics and Heat Transfer Division, Department of Mechanical Engineering, University of Minnesota, 111 Church Street, S.E., Minneapolis, Minnesota 55455. Responsible person, Roy C. Tew, Jr., organization code 5490, 216–433–8471.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category: 34 Distribution: Nonstandard
Available electronically at http://gltrs.grc.nasa.gov/GLTRS
This publication is available from the NASA Center for AeroSpace Information, 301–621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

NASA Glenn Research Center (GRC) is funding Cleveland State University (CSU) to develop a reliable Computational Fluid Dynamics (CFD) code that can predict engine performance with the goal of significant improvements in accuracy when compared to one-dimensional (1-D) design code predictions. The funding also includes conducting code validation experiments at both the University of Minnesota (UMN) and CSU. In this paper a brief description of the work-in-progress is provided in the two areas (CFD and Experiments). Also, previous test results are compared with computational data obtained using (1) a 2-D CFD code obtained from Dr. Georg Scheuerer and further developed at CSU and (2) a multidimensional commercial code CFD-ACE+. The test data and computational results are for (1) a gas spring and (2) a single piston/cylinder with attached annular heat exchanger. The comparisons among the codes are discussed. The paper also discusses plans for conducting code validation experiments at CSU and UMN.

14. SUBJECT TERMS
Stirling cycle; Stirling engines; Simulation; Fluid mechanics; Heat transfer

15. NUMBER OF PAGES
13

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	